Preliminary CFD Analysis for a Thermal Mixing Test by a Coupled Solver

H. S. Kang,[a] Y. S. Kim,[a] H. G. Jun,[a] C. H. Song [a]
a Korea Atomic Energy Research Institute, P. O. Box 105,Yuseong,Taejon, Korea, 305-600, hskang3@kaeri.re.kr

1. Introduction

A CFD sensitivity analysis for a thermal mixing test [1,2] simulating the DCC (Direct Contact Condensation) of a steam jet in a subcooled water pool was performed to find an optimized CFD calculation methodology [3]. In the previous CFD analysis, CFX-4.4 using the SIMPLE algorithm was used for a transinet calculation of 30 seconds. A long computation time was needed for CFX-4.4 calculation compared to the coupled solver of CFX-10.0 because the parallel computation environment of CFX-4.4 was worse than that of CFX-10.0 [4]. And also, much more the iteration numbers were necessary at a pressure correction step in the CFX-4.4 analysis to have a converged solution when compared to that of CFX-10.0. Therefore, the coupled solver of CFX-10.0 was introduced to speed up the computation time because the developed numerical methodology for a thermal mixing should be applied for a long accident analysis of about 2000 seconds for a safety assesment of APR 1400 [2].

2. Thermal Mixing Test [1]

The thermal mixing test was performed by changing the steam mass flux and the tank water temperature in the transient and the quasi steady states. Eight thermocouples to measure the temperature of the steam and the entrained water flowing into the steam were installed in the tank, and two measurement rigs of 27 thermocouples were installed to obtain the thermal mixing pattern. A second rig was installed to observe the extent of the thermal mixing along the circumferential direction in the tank. In the case of the high steam mass flux, the thermal mixing phenomena in the tank showed a nearly axis symmetric pattern.

3. CFD Analysis

3.1 Flow Field Models and Boundary Conditions

In the test, the discharged steam from a sparger flowed into the water as a jet flow, and then it was quickly condensed to water by the DCC [2]. The steam condensation region model for the DCC phenomenon was used [2]. Thermal mixing phenomenon in the water tank was treated as an incompressible flow, a free surface flow of air between the water, a turbulent flow, and a buoyancy flow. The governing equations used in this study were the Navier-Stokes and the energy equations with a homogenous multi-fluid model [2,3]. Turbulent flow was modeled by the standard k-ε turbulent model, and the buoyancy was modeled by the Boussinesq approximation. The inlet boundary condition was set at the end of the steam condensation region with a time dependent velocity and temperature. The pressure outlet boundary conditions were set for the tanks upper region which was extended upward by 0.5m to move the fully developed condition imposed by applying the pressure outlet condition into the downstream of the flow field. The outlet conditions for the entrained water were applied to the upper and lower region of the steam condensation region by a negative value of the velocity with the inlet condition in the CFX4.4 and CFX-10.0.

3.2 Grid and Numerical Models for Sensitivity Analysis

A multi-grid with an axis symmetric condition simulating the sparger and the subcooled water tank for the CFD calculation was generated (Fig. 1). The axis symmetric model was introduced because the flow pattern in the tank was estimated to vary a little in the circumferential direction and it could reduce the computational time. The sensitivity calculation of the mesh distribution and the numerical method were performed (Table 1). Three cases only used different mesh distributions in the grid model by using the same upwind 1st method for a convection term discretization. In case 1, a total of 9,588 cells were generated, and the first grid from the right wall was located at the position of 100~300 of y+. In the second case for the sensitivity study, the mesh distribution was rearranged based on the comparison results between the CFD data of case 1 and the test data. A total of 23,835 cells and 12~50 of y+ were generated to predict the temperature close to the test data even though the computation time took longer than that of case 1. Especially, more meshes were distributed at around the jet flow and a region near to the wall. Case 3 grid model had 31,020 cells and 12~50 of y+. The meshes of case 3 were more densely located at the transition region in the upper region of the tank than those of the case 2. In case 4 and 5, the same mesh distribution of case 1 was used whereas the numerical model of the convection term discretization was changed to a QUICK scheme and a High Resolution scheme implemented in CFX-10.0, respectively [4].

Table 1. Sensitivity Calculation Conditions

	Cell No.	Horizontal × Vertical	Convection Term Discretization
Case 1	9,588	63×160	Upwind 1^{st}
Case 2	23,835	103×263	Upwind 1^{st}
Case 3	31,020	113×273	Upwind 1^{st}
Case 4	9,588	63×160	QUICK
Case 5	9,588	63×160	High Resolution [4]

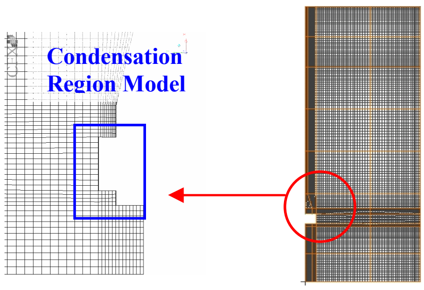

Figure 1. Grid model and the mesh distribution

3.4 Discussion on the CFX Results

The temperature comparison of the test data with the CFX results for 30 seconds except Case 5 at 3 thermocouple locations are shown in Fig. 2. The comparison of the CFD results with the test data showed a good agreement within 7~8% value [2,3]. This difference may have arisen from the fact that the temperature and the velocity of the calculated condensed water by the condensation region model were higher than the real value. Another reason may be a limitation of the condensation region model by using the area average concept. The sensitivity calculation results of CFD were very similar to each other at the region (TC706) between the sparger and the tank wall irregardless of the cases. However, the CFD sensitivity results showed a small temperature distribution difference at the upper (TC729) and lower region (TC728) where the condensed water jet arrived after colliding with the tank wall. Especially for the high upper region, case 4 using the QUICK scheme predicted the test data better than the other cases using the Upwind scheme. The results of Case 5 were very similar to Case 4 even though Case 5 was only calculated for 10 seconds. This is because the high resolution scheme of Case 5 is analogous to the QUICK method. As for a comparison of the computation time of Case 5 against other cases, the easy parallel environment of CFX-10.0 greatly reduced the calculation time even though a quantitative comparison was impossible because Case 5 was performed by 2 or 4 cpus. However, the results of a free surface behavior by CFX-10.0 should be carefully investigated.

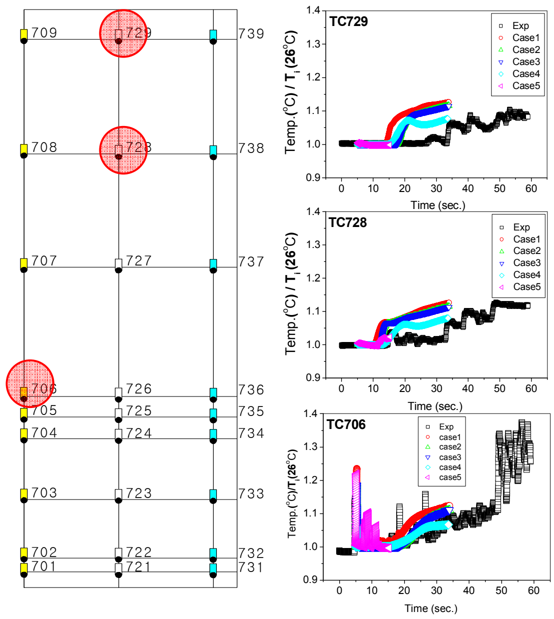

Figure 2. Temp. distribution of CFD and Test results

4. Conclusion and Further Research

From the results of the sensitivity analysis, it was known that the numerical methods, the grid meshes distribution and CFD solvers affected the temperature distribution of the thermal mixing and the computation time. Therefore, it is believed that this sensitivity calculation results may assist in the establishment of the strategy for the CFD analysis of APR 1400 IRWST Pool.

ACKNOWLEDGEMENTS

This work was financially supported for the nuclear R&D program from the Ministry of Science and Technology of Korea.

REFERENCES

[1] Y. S. Kim et al., “Steam Condensation Induced Thermal Mixing Experiment Using B&C Facility”, Technical Report, KAERI/TR-2933/2005, KAERI, 2005.
[2] H. S. Kang et al., “A CFD Analysis of Characteristics of the Thermal Mixing Under the Transient of the Steam Discharge in a Subcooled Water Tank”, Technical Report, KAERI/TR-3008/2005, KAERI, 2005.
[3] H. S. Kang et al., “A CFD Analysis for Thermal Mixing in a Subcooled Water Tank under Transient Steam Discharge Conditions”, *J. of Computational Fluids Engineering*, **vol. 11**, **No. 2**, 2006.
[4] Ansys, Inc., “CFX-10.0 Manual”, 2006.